

Dynamic and Impact Analysis of Aerospace Vehicles Using ABAQUS/Explicit

Presented at the
2004 FEMCI Workshop
NASA/GSFC, Greenbelt, MD

Kyle C. Indermuehle
Product Management – Aerospace Applications
ATA Engineering -- ABAQUS

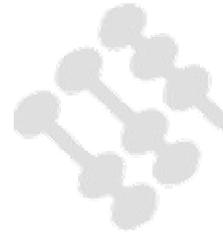
Aerospace Vehicles are Complex Systems With Numerous Different Analyses that Need to Be Performed



- **Aerospace systems are exposed to various loading conditions that all need to be fully analyzed**
 - Static, dynamic, thermal, acoustic, operational
- **Typical satellite analyses include:**
 - Dynamic analysis for shipping, launch, and operation
 - Detailed component stress and margin calculations
 - Mechanism analysis for deployment of solar panels and reflector
 - Thermal analysis for in-orbit operation
- **These analyses are typically performed in the linear domain**
 - Often the modal domain for dynamic problems
- **But, there are some load cases that cannot be analyzed linearly...**

Impact is a Highly Nonlinear, Dynamic Event

- **Example problem: Satellite impact with ground**

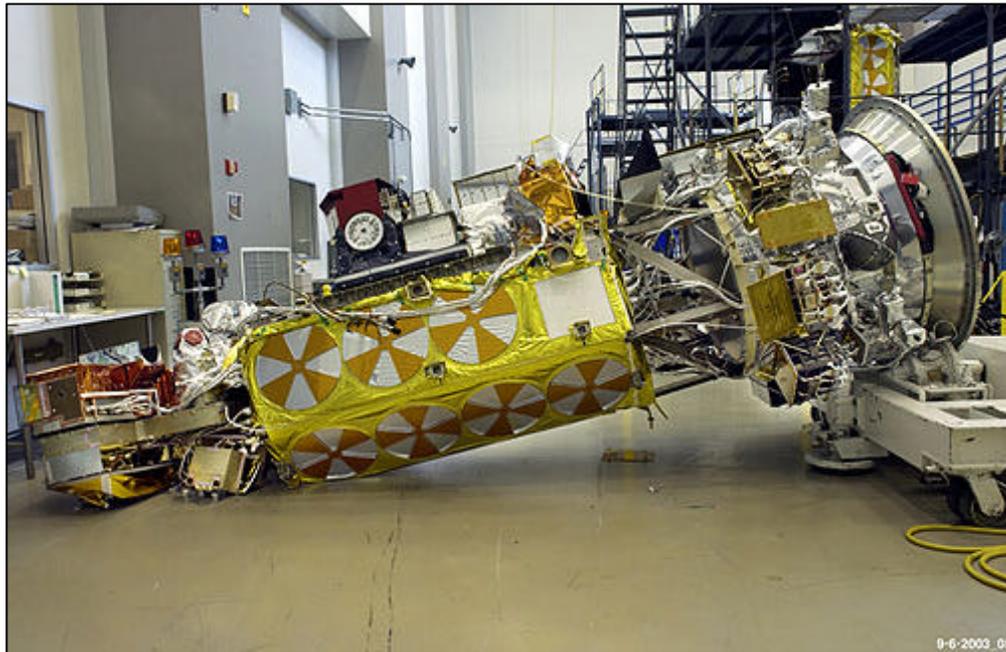


Engineering Challenges

- **Complicated, nonlinear, dynamic event with many contact regions and damage / failure**

Goals of analysis

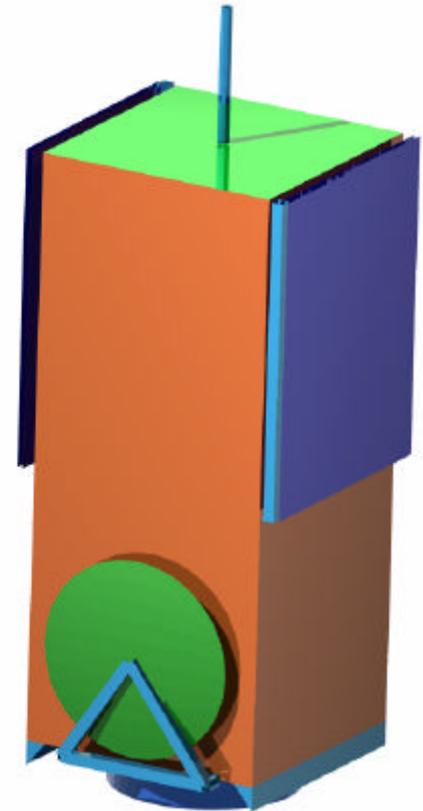
- **Determine forces on satellite caused by impact**
- **Determine if components / joints failed and correlate to actual results**
- **Determine peak component accelerations**



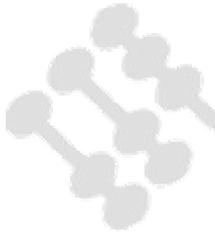
Source: Aviation Week & Space Technology

Disclaimer

- **This paper is a discussion of methodology and the latest simulation capabilities**
- **The impact event is used simply as an example**
 - Simulation is based solely on information implied from the image in AWST
 - A simple, generic satellite is used for the analysis
- **The methodology and capabilities discussed have been used on other similar analyses**



Methodology for Impact Analysis is to Start Simple and Add Increasing Complexity



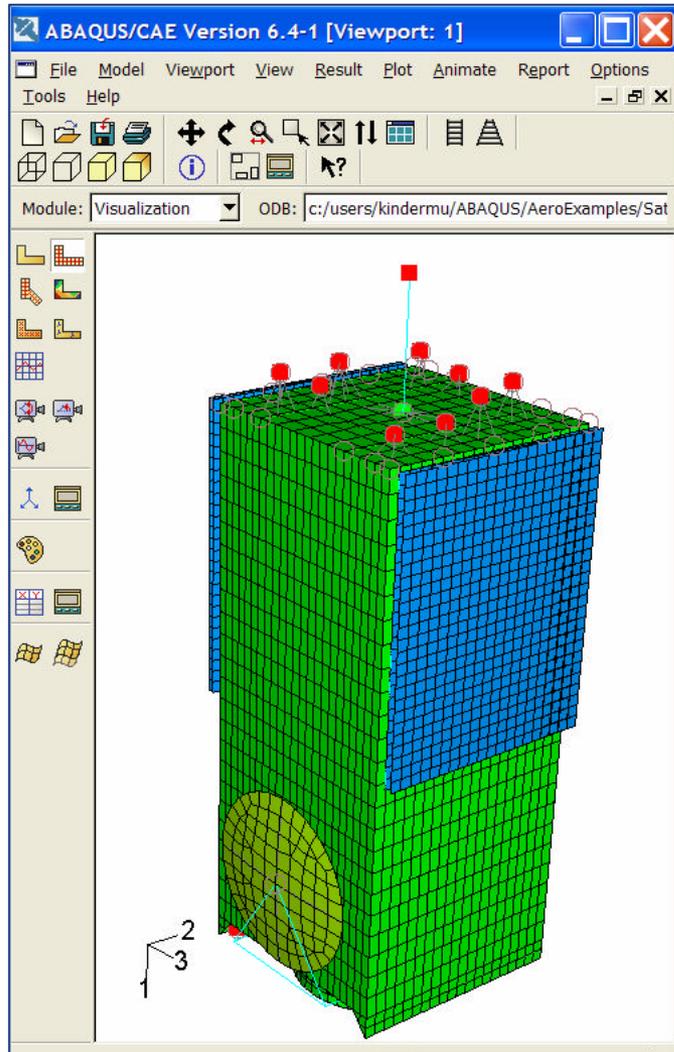
- **Methodology**

- Perform simple rigid body / mechanism dynamic analysis
 - Allows for quick insight into event
- Perform flexible body impact dynamic analysis
 - More accurate simulation of true event
- Perform flexible body impact analysis with failure models
 - Accounting for joint / material failure further increases accuracy of simulation

- **Workflow**

- Translate NASTRAN FEM to ABAQUS
- Perform rigid body / mechanism dynamic analysis
- Perform flexible body dynamic analysis using ABAQUS /Explicit
- Perform flexible body analysis with failure using ABAQUS /Explicit

NASTRAN Models Can Easily Be Translated into ABAQUS Using *ABAQUS fromnastran* Utility

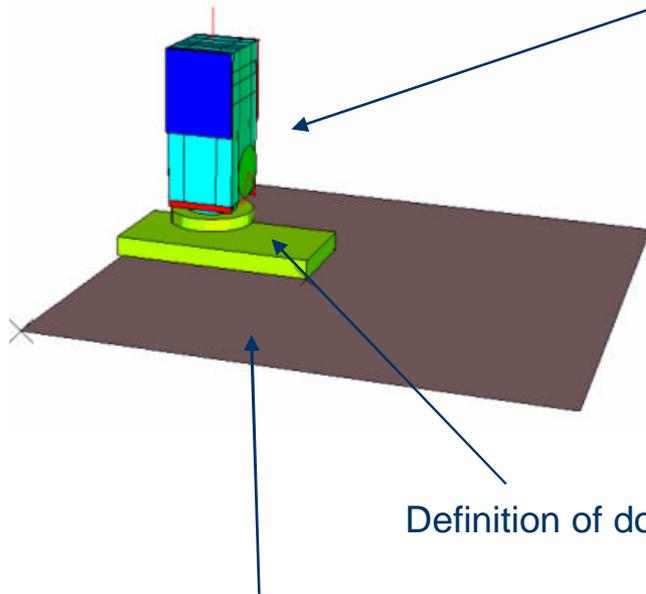
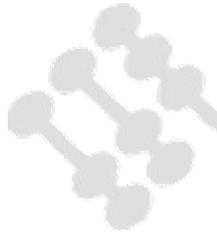


NASTRAN Bulk Data processed: (no errors in translation)

CBAR, CONM2, CORD2R, CQUAD4, CTRIA3, GRID,
MAT1, MAT2, PBAR, PSHELL, RBAR, RBE2, RBE3

Mode	ABAQUS	Nastran CQUAD4	Difference	Nastran CQUADR	Difference
1	11.00	10.66	-3.2%	10.82	-1.7%
2	11.05	10.71	-3.2%	10.86	-1.7%
3	17.39	16.43	-5.9%	16.63	-4.6%
4	18.53	17.56	-5.5%	17.69	-4.7%
5	28.42	28.16	-0.9%	28.31	-0.4%
6	28.54	28.29	-0.9%	28.44	-0.4%
7	33.08	31.33	-5.6%	32.30	-2.4%
8	33.36	31.38	-6.3%	32.63	-2.2%

Impact Analysis: Step 1—Prepare Model



Definition of contact surfaces

```

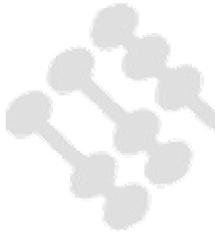
*surface, name=ssat, type=node
  sat_box
*surface, name=sground, type=element
  ground, sneg

*contact pair, cpset=ct_sat, interaction=sat_int, mechanical constraint=penalty
  ssat, sground
*surface interaction, name=sat_int
*friction
  0.8,
*contact damping, definition=damping coefficient
  0.05,
*contact controls, cpset=ct_sat
  
```

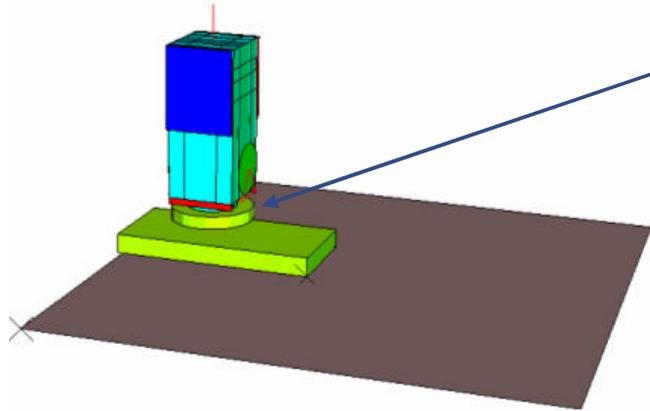
Definition of dolly and mount

```

** Rigid ground
*NODE, NSET=ALLNODES, SYSTEM=R
  58179, 10.000000E+01, -2.500000E+02, -2.200000E+02
  58180, 10.000000E+01, 5.000000E+01, -2.200000E+02
  58181, 10.000000E+01, -2.500000E+02, 8.000000E+01
  58182, 10.000000E+01, 5.000000E+01, 8.000000E+01
*ELEMENT, TYPE=S4R, ELSET=ground
  56720, 58179, 58180, 58182, 58181
*rigid body, ref node = 58180, elset=ground, position=inp
  
```

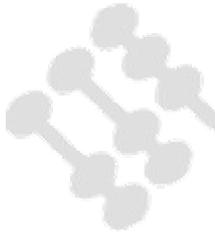


Impact Analysis: Step 2—Define Event Excitation



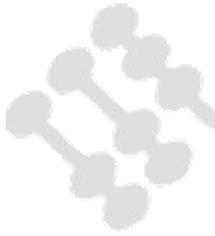
Simulation is of rotation of mounting plate

```
*step
*dynamic, explicit
, 10.0
** fix ground, dolly, and mount
*boundary
58180,1,6,0.0
48480,1,6,0.0
48326,1,5,0.0
**define rotation
*boundary, amplitude=rotation, type=displacement
48326, 6, 6, 1.0
*amplitude, name=rotation, definition=tabular
0.0, 0.0, 3.0, 0.4
*dload
all_massive_elements, grav, 386.088, 1.0, 0.0, 0.0
*end step
```



Impact Analysis: Step 3—Run Simulation

- **Model is ready to run**
 - Full FEM translated from NASTRAN
 - Assumption is that this is a legacy model
 - Rigid ground, dolly, and mount defined
 - Contact surfaces and surface friction defined
 - Event excitation defined
- **Initial estimation of stable time increment is 1e-8 seconds**
 - For 10-second simulation this means 1e9 time steps
 - Run will take 4+ hours



Impact Analysis: Step 3a—Run Rigid Body Simulation

- **Define satellite as a rigid body**

```
*rigid body, ref node=27329, elset=sat
```

- **Define /Explicit analysis**

```
*dynamic, explicit, direct user control  
0.01, 10.0
```

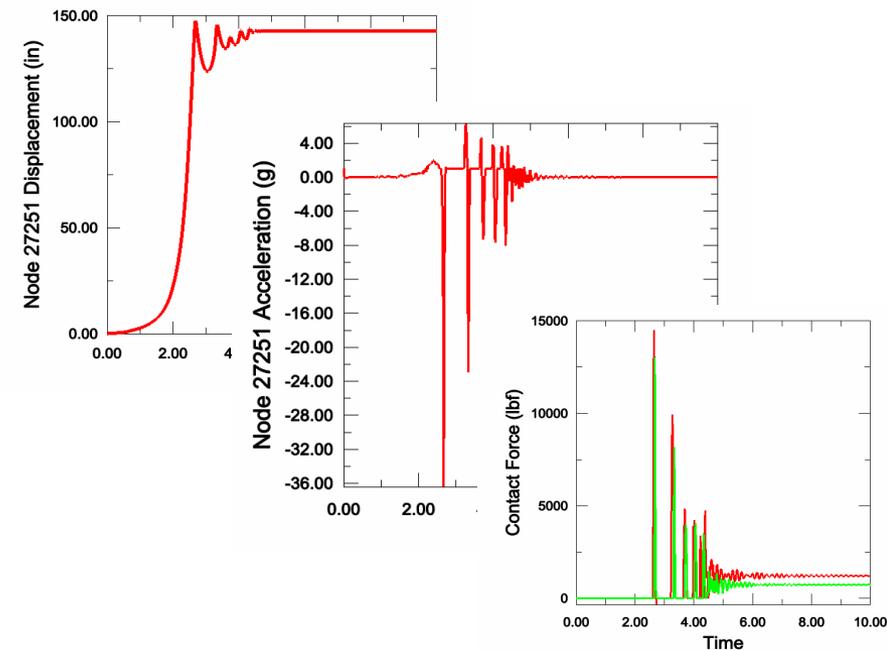
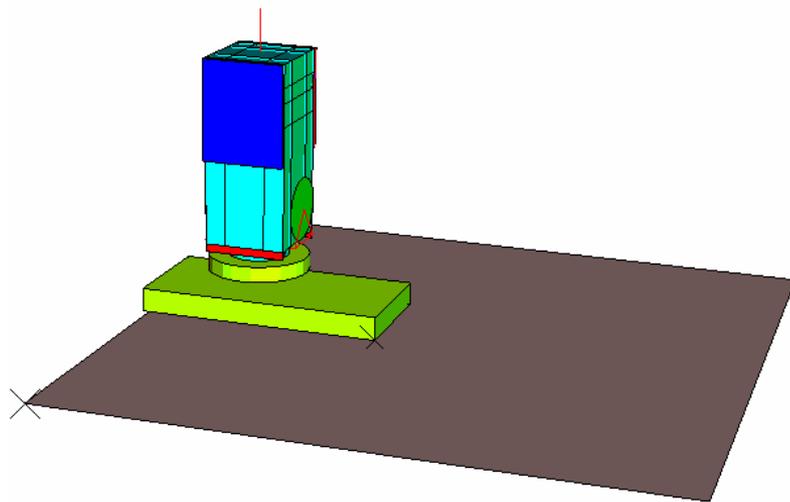
- **Run rigid body simulation**

- Time increment is now 1e-2 seconds
- For 10-second simulation this means 1e3 time steps
- Run will take 1 minute

```
abaqus job=rigid_body double
```

Impact Analysis: Step 3a—Postprocess Rigid Body Simulation

- **Rigid body analysis provides quick insight into the event**
 - *RIGID BODY makes FE mesh a rigid body
 - Fast run time (1 minute for this model on a laptop)
 - Easy to verify and debug model
 - Provides insight such as displacement, acceleration, contact forces



Impact Analysis: Step 3b—Run Flexible Body Simulation

- **Remove rigid body definition for satellite**

```
**rigid body, ref node=27329, elset=sat
```

- **Define ABAQUS/Explicit analysis**

```
*dynamic, explicit  
, 10.0
```

- **Run flexible body simulation**

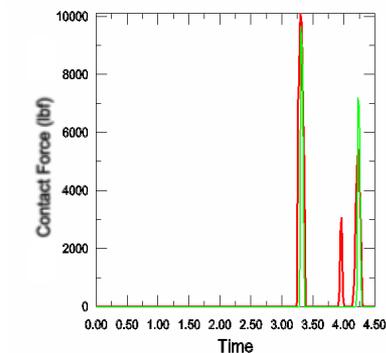
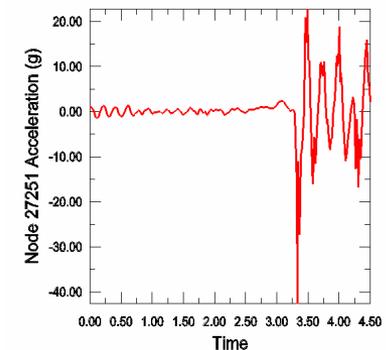
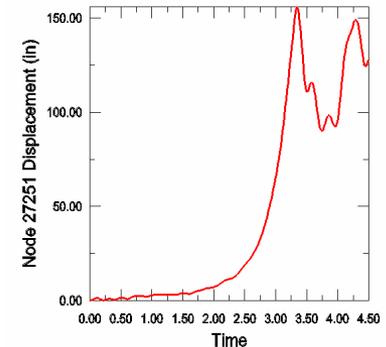
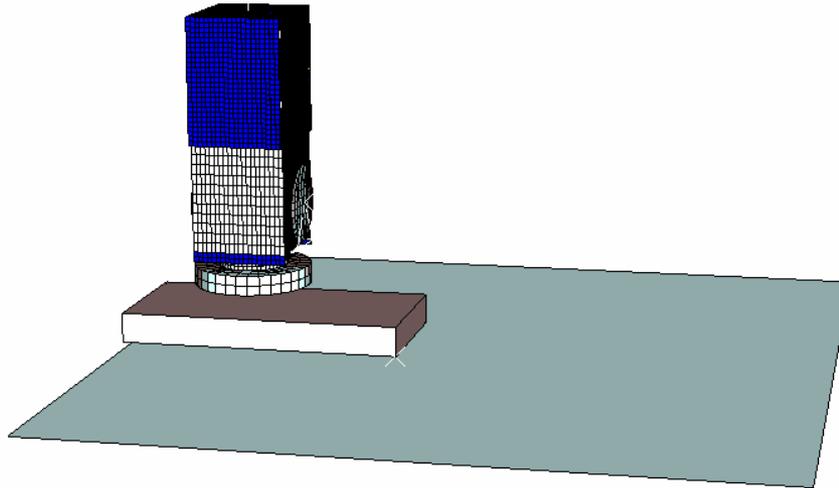
- Time increment is now 1e-8 seconds
- For 10-second simulation this means 1e9 time steps
- Run will take 4+ hours
- Parallel processing can be used to reduce run to 2 hours

```
abaqus job=rigid_body cpus=2
```

Impact Analysis: Step 3b—Postprocess Flexible Body Simulation

- **Flexible body simulation**

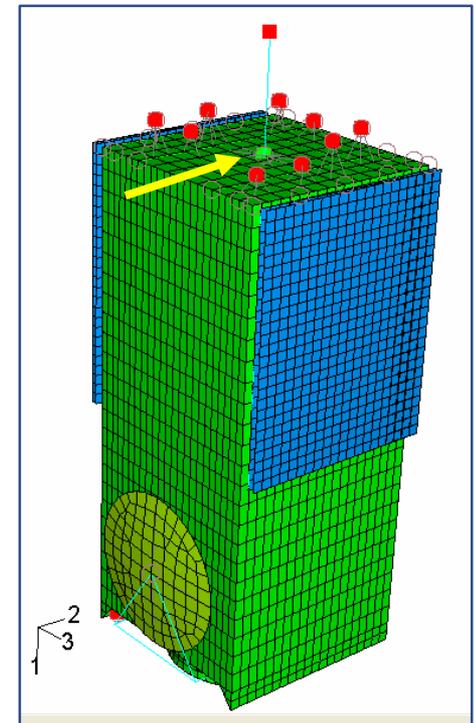
- Same model; run file as rigid body analysis, just removed *RIGID BODY from input file
- Analysis time now over 4 hours on 3 GHz PC for half of the event
 - Time reduced to 2 hours using parallel processing
- Can recover displacement, acceleration, contact forces, element forces and stresses



Impact Analysis: Step 3c—Run Flexible Body Simulation with Component Failures

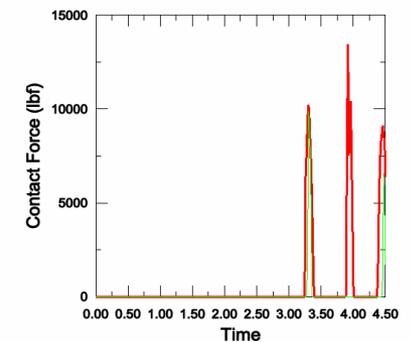
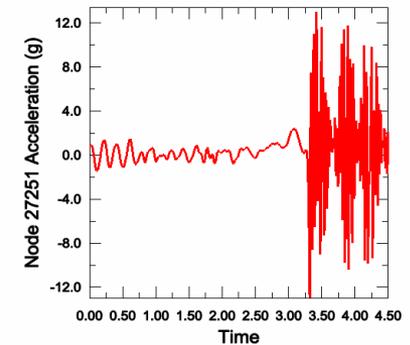
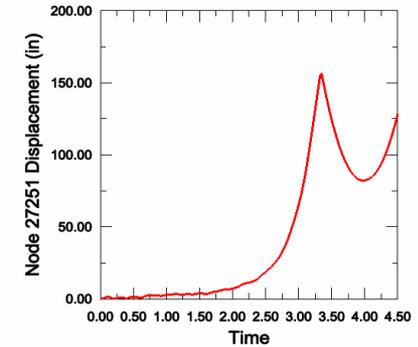
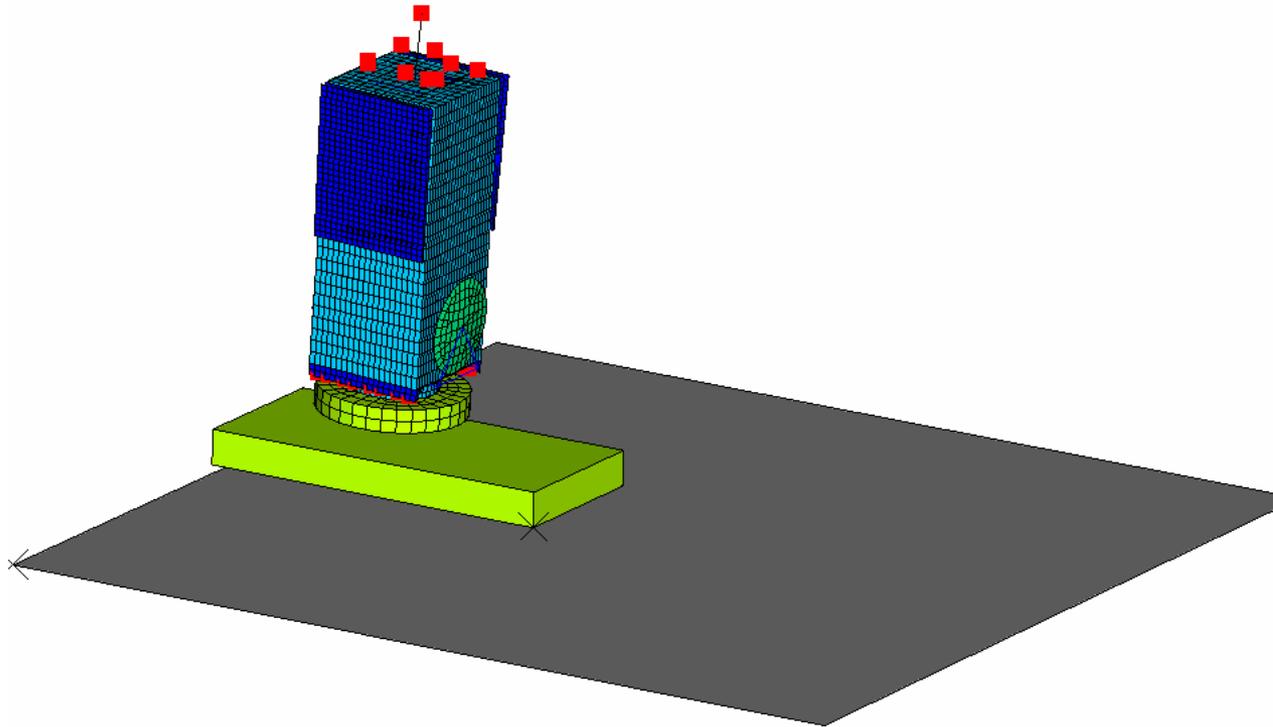
- Add ***CONNECTOR FAILURE** to connector definitions

```
**connector element connection 27252 to 127252 (antenna mass joint)
*element, type=conn3d2, elset=antenna_joint
  127252, 127252, 27252
*connector section, elset=antenna_joint, behavior=antenna_behav
  weld
*connector behavior, name=antenna_behav
*connector failure, component=1, release=all
  ,,-5000,5000
*connector failure, component=2, release=all
  ,,-5000,5000
*connector failure, component=3, release=all
  ,,-5000,5000
*connector failure, component=4, release=all
  ,,-5000,5000
*connector failure, component=5, release=all
  ,,-5000,5000
*connector failure, component=6, release=all
  ,,-5000,5000
```



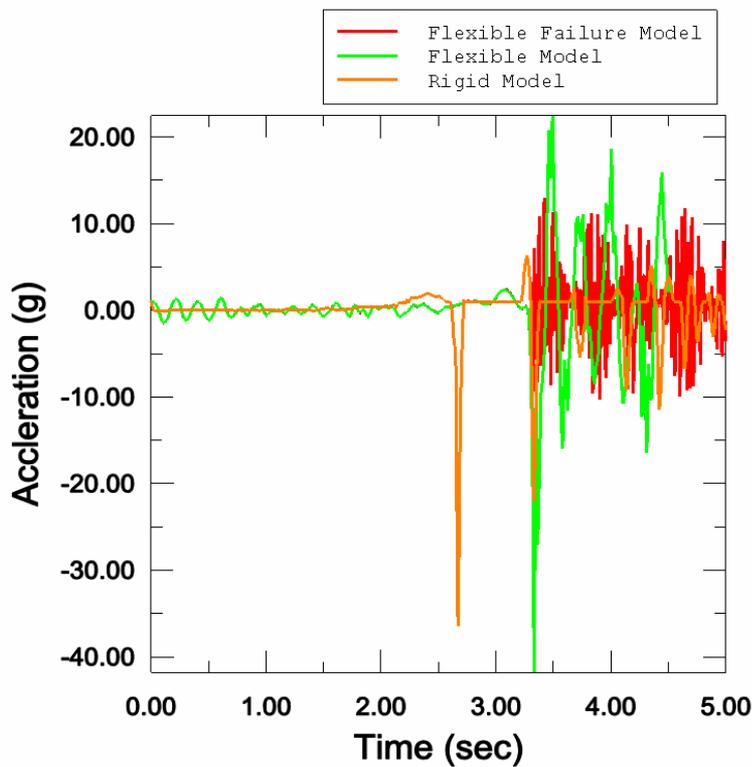
Impact Analysis: Step 3c—Postprocess Flexible Body Simulation with Component Failures

- **Failure models include**
 - Force overload, peak displacement, material plasticity, laminate failure, ABAQUS user subroutines
- **Simulation accurately reflects the change in the structure**

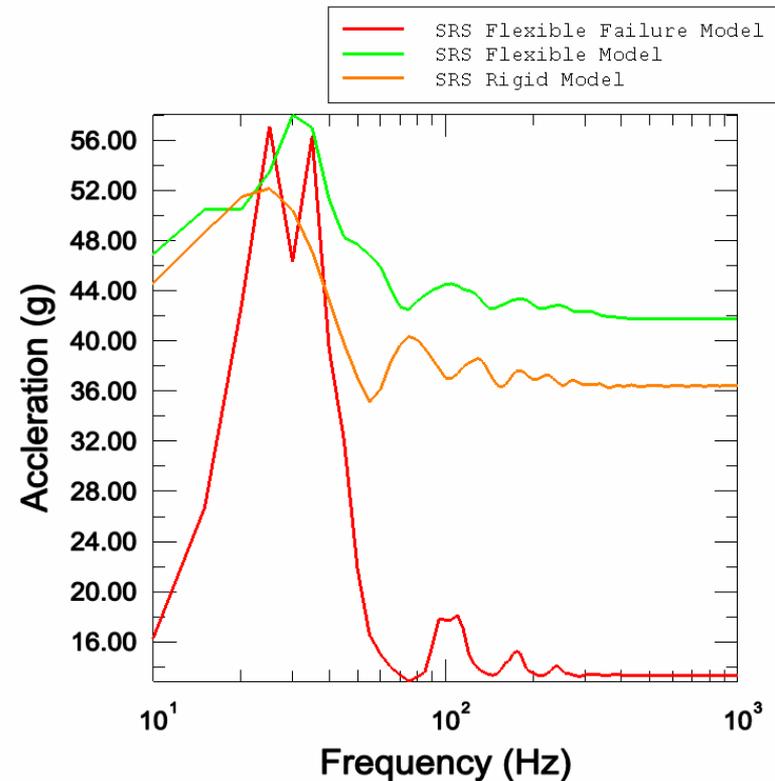


Comparison of Rigid Body, Flexible, and Flexible with Failure Shows Flexible Model Accuracy

- Comparison of responses shows flexible body model has responses 10% higher than rigid body



Time response



Shock response spectra

Methodology Used for Analysis of Satellite Impact Was to Start Simple and Add Increasing Complexity



- **Methodology**

- Use existing loads / dynamics model for analysis
 - Can translate from NASTRAN using *fromnastran* utility
- Define impact analysis
 - Define ground, dolly, and mount (rigid)
 - Define contact surfaces
- Perform simple rigid body/mechanism dynamic analysis
 - Allows for quick insight into event
- Perform flexible body impact dynamic analysis
 - More accurate simulation of true event
- Perform flexible body impact analysis with failure models
 - Accounting for joint/material failure further increases accuracy of simulation

Simulation of Satellite Impact for Varying Model Fidelity Allows Progressive Accuracy and Insight into Event

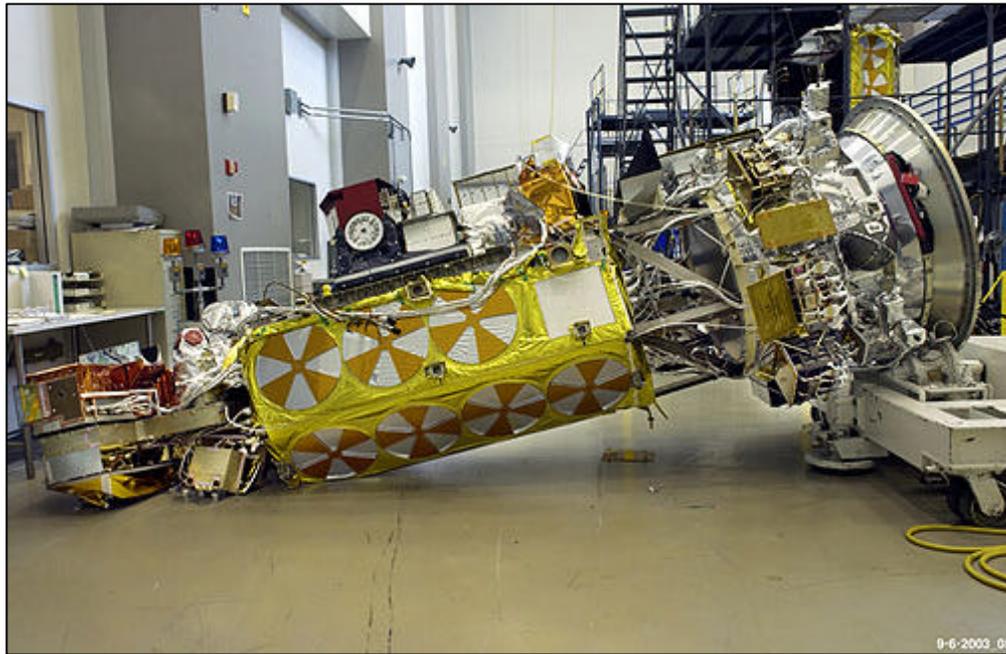


Engineering Challenges

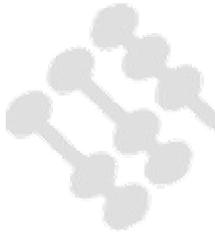
- **Complicated, nonlinear, dynamic event with many contact regions and damage / failure**

Simulation Capabilities

- **Able to use existing FE model for analysis (typically a Nastran CLA model)**
- **Easily change from flexible to rigid body analysis**
- **Robust, general contact algorithm**
- **Nonlinear material properties and failure criteria**
- **Parallel processing to reduced run time**
- **Mechanism – flexible body co-simulation**



Current Software Technology Provides the Capability to Perform Multiple Simulations in One Toolkit



- **Unified FEA**

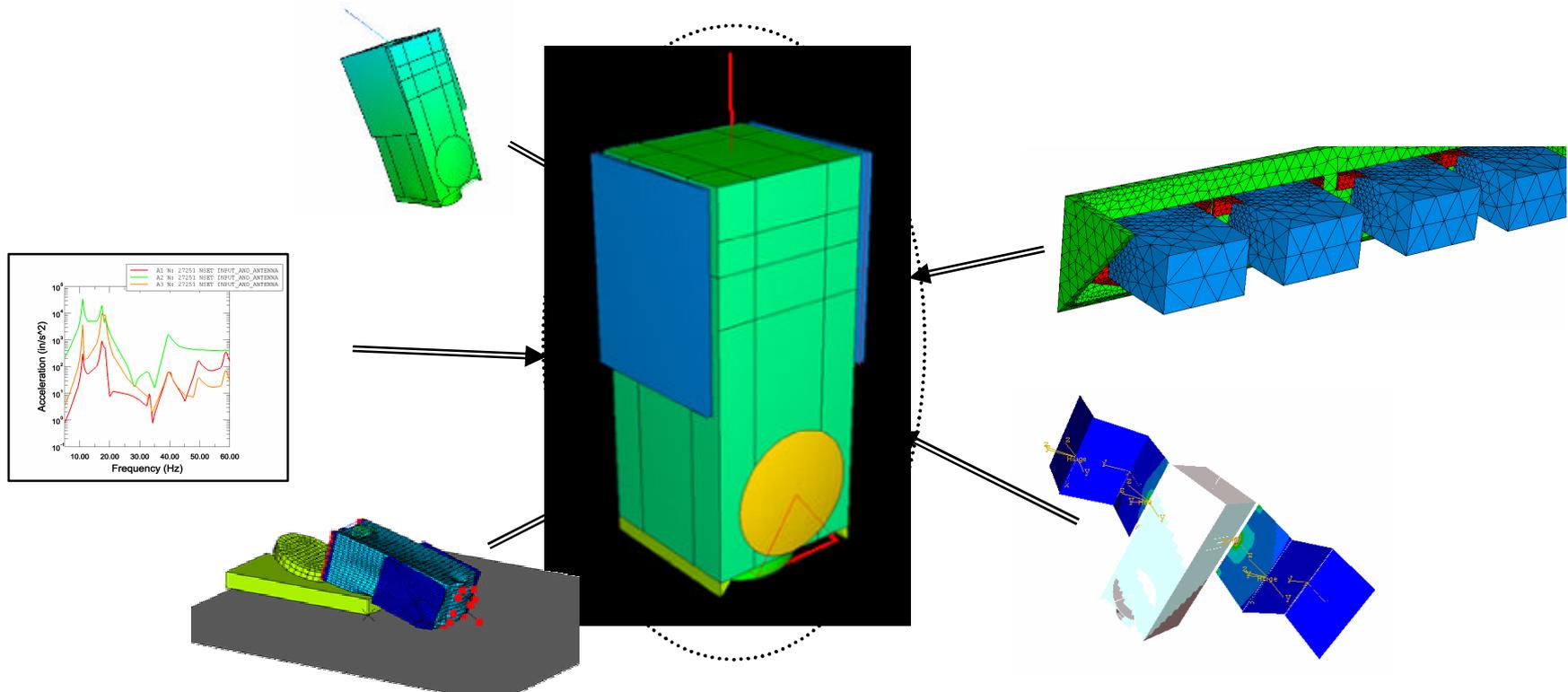
- Fewer software products needed
- More and smarter reuse of models and results
- Better technical solution through coupled analysis
- Reduced data management

- **One FE model and one software code to perform**

- Dynamic analysis
- Nonlinear static analysis
- Mechanism simulations
- Impact/crash
- Structural-thermal coupled problems

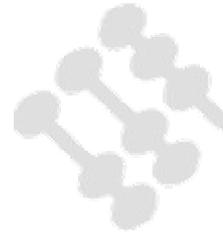
Satellite Unified FEA

- **Global dynamic, component-level stress, mechanism, and impact analysis can all be performed using**
 - One code—ABAQUS
 - One FE model—With minor changes (*RIGID BODY, *SUBMODEL, *COMPONENT FAILURE)



Satellite Dynamic Analysis

- **Example problem: Analysis of launch vehicle loads on satellite**

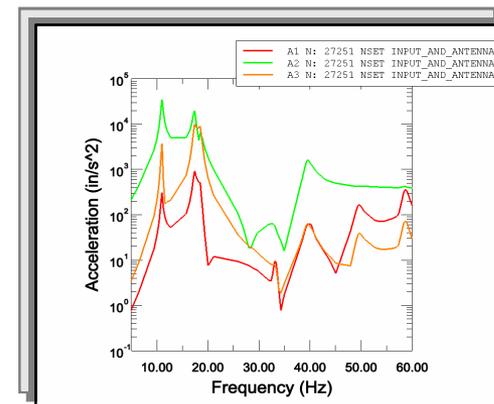
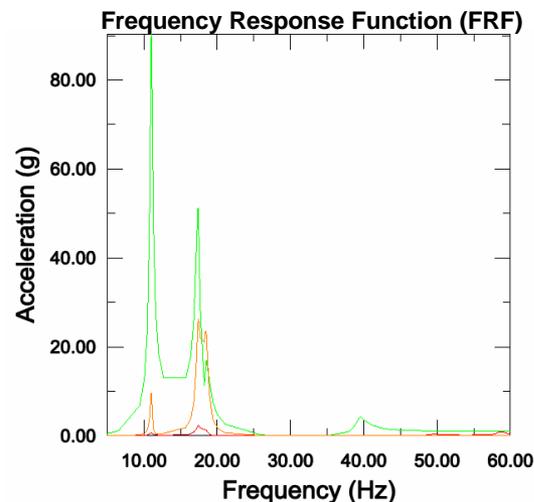
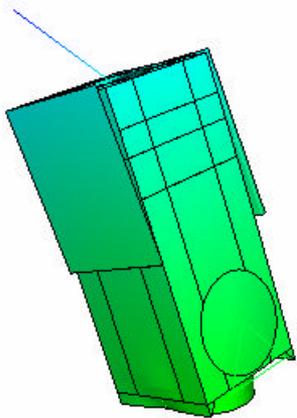


Engineering challenges

- Solving for modes of a complicated often large FE model
- Definition of dynamic environment
- Output of many responses
- Graphically viewing responses

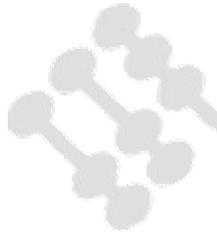
ABAQUS solutions

- Efficient Lanczos solver
- Straight forward definition of excitation environment
- ELSET and NSET definition for groups of output entities
- Postprocessing is easy using ABAQUS/Viewer



Satellite Component Analysis

• Example problem: Stress analysis of brackets using Submodeling

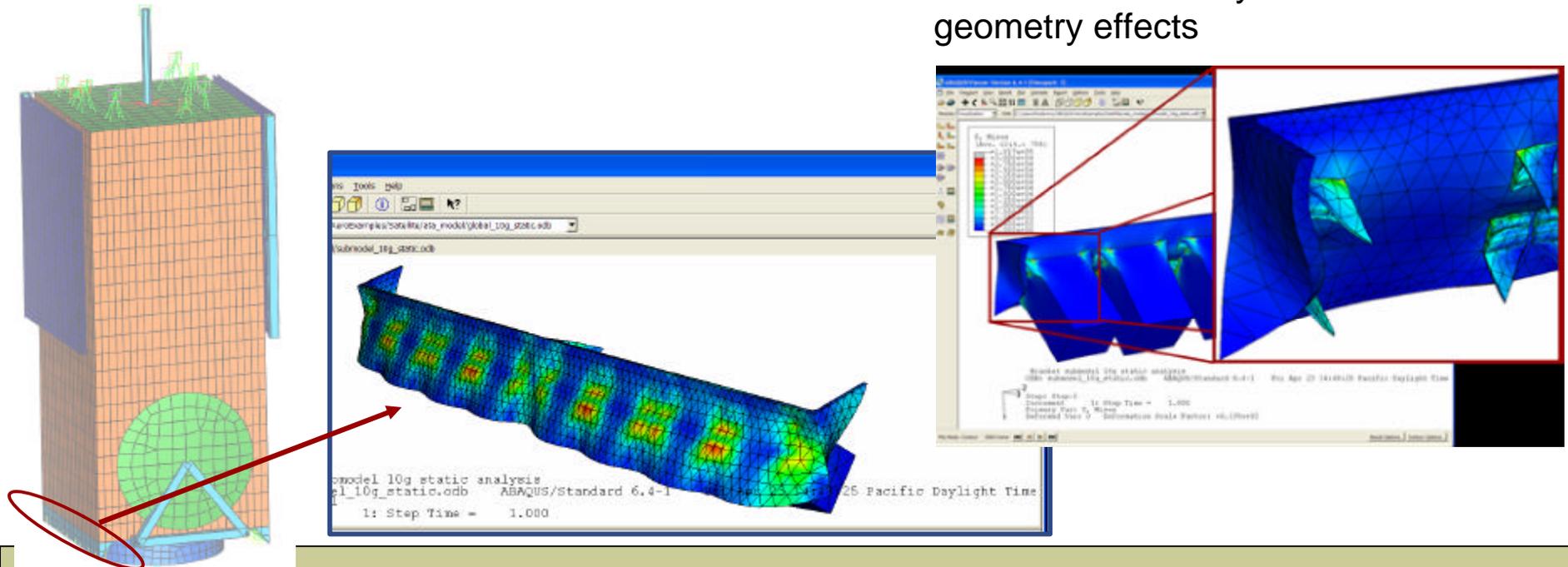


Engineering challenges

- Multiple static load cases
- Possible material nonlinearities
- Thermal loads
- Easy, visual postprocessing of results

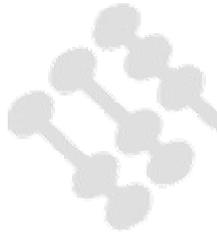
ABAQUS solutions

- Submodeling capability for easy analysis
- Can efficiently analyze many load cases using perturbation analysis
- Advanced FEA capabilities include material nonlinearity and nonlinear geometry effects



Satellite Mechanism Analysis

- **Example problem: Deployment of solar panels**

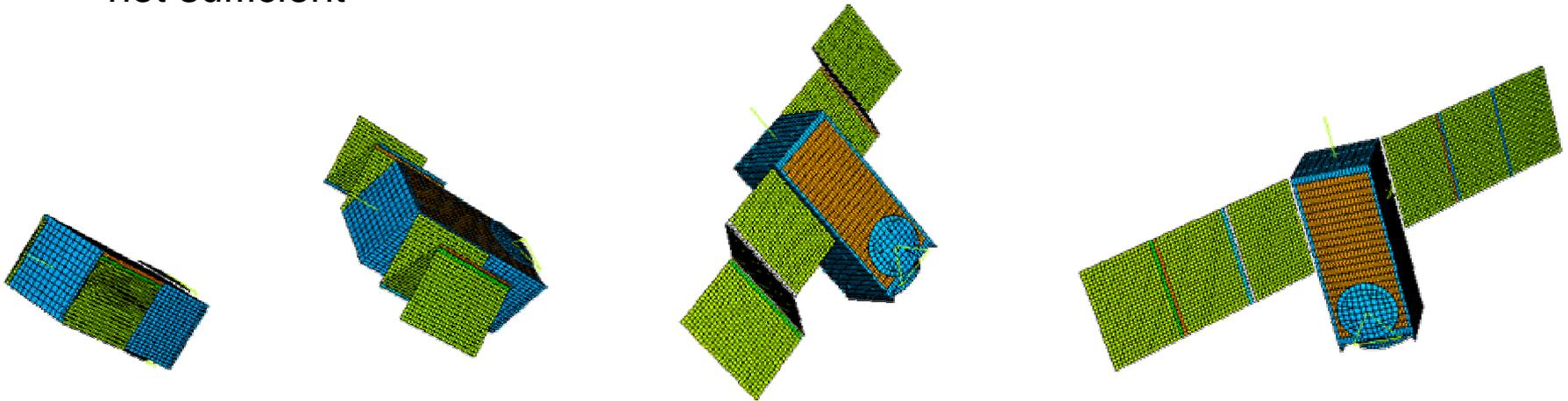


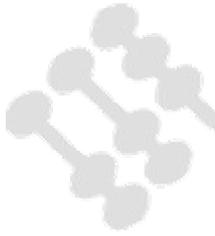
Engineering challenges

- Mechanism analysis
- Need to understand forces and stresses due to deployment
- Flexibility of panels is important to analysis—rigid body simulation is not sufficient

ABAQUS solutions

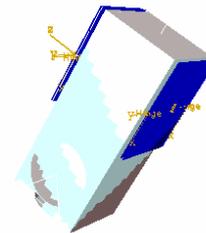
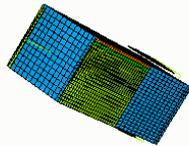
- ABAQUS can solve the coupled mechanism-flexible body problem, including nonlinear effects





Satellite Mechanism Analysis

- **Animation of deployment**



Dynamic and Impact Analysis of Aerospace Vehicles using ABAQUS/Explicit

Presented at the
2004 FEMCI Workshop
NASA/GSFC, Greenbelt, MD

Kyle C. Indermuehle
Mike Sasdelli

ATA Engineering / ABAQUS
ABAQUS East

858.792.3958
410.420.8587